Hypersonic CFD Space Shuttle Simulations

Space Operations Mission Directorate

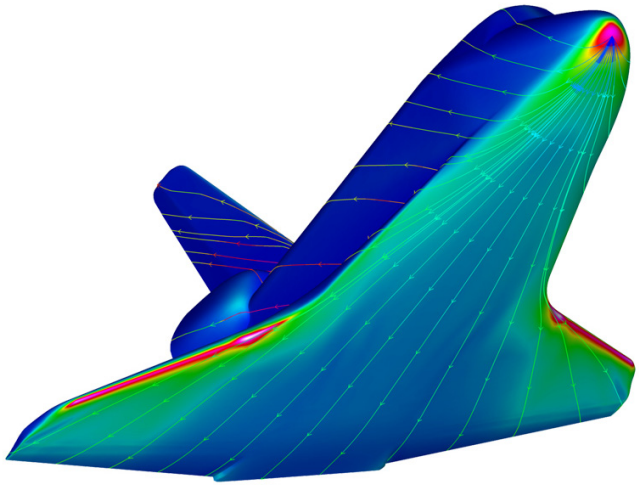
Computational fluid dynamics (CFD) simulations provide high-fidelity assessment of possible damage to the Space Shuttle, in support of the shuttle's Damage Assessment Team. Using the powerful and reliable computing resources at the NASA Advanced Supercomputing (NAS) facility, researchers are able to employ these simulations in near real-time to assess damage and repair scenarios during shuttle operation.

Rapid hypersonic CFD simulations of damaged and repaired regions are increasingly important for real-time risk assessment during vehicle operations. Localized CFD simulations allow researchers to investigate small features on the shuttle, while leveraging existing high-fidelity simulations completed before the missions. This technique provides rapid turnaround during missions, producing results within hours instead of days or weeks.

Parallel CFD codes, such as NASA Ames Research Center's DPLR (Data Parallel Line Relaxation) flow solver, are used to understand thermal protection system damage, as well as to develop flight experiments and confidently design the next generation of reentry vehicles. Additionally, NAS resources are used throughout the year to develop new simulation capabilities and help design future flight experiments.

Recently, the Space Shuttle Program approved the installation of small, instrumented protuberances on the underside of the shuttle. Data gathered from these instruments during recent shuttle missions will help engineers better characterize chemically reactive high-speed flows, both for remaining shuttle flights and future reentry vehicles.

Todd White, NASA Ames Research Center
todd.r.white@nasa.gov



DPLR computational fluid dynamics simulation of instantaneous heating on the Space Shuttle. *Todd White, NASA/Ames*